Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Pad Shape	Hole Length
□	30	7.87mil (0.200mm)	PTH	Round	TOP - BOTTOM	Rounded	-
⊠	4	8.00mil (0.203mm)	PTH	Round	TOP - BOTTOM	Rounded	-
B	7	12.10mil (0.307mm)	PTH	Round	TOP - BOTTOM	Rounded	-
⊕	811	12.20mil (0.310mm)	PTH	Round	TOP - BOTTOM	Rounded	-
⊠	1	17.72mil (0.450mm)	PTH	Round	TOP - BOTTOM	Rounded	-
D	2	23.62mil (0.600mm)	PTH	Slot	TOP - BOTTOM	Rounded	51.18mil (1.300mm)
⌘	3	31.50mil (0.800mm)	PTH	Rectangle	TOP - BOTTOM	Rounded Rectangle	78.74mil (2.000mm)
▽	2	31.50mil (0.800mm)	PTH	Rectangle	TOP - BOTTOM	Rounded Rectangle	90.55mil (2.300mm)
◇	2	33.47mil (0.850mm)	PTH	Round	TOP - BOTTOM	Rounded	-
✡	4	40.00mil (1.016mm)	PTH	Round	TOP - BOTTOM	(Mixed)	-
F	2	40.16mil (1.020mm)	NPTH	Round	TOP - BOTTOM	Rounded	-
E	3	40.16mil (1.020mm)	PTH	Round	TOP - BOTTOM	(Mixed)	-
◙	3	47.24mil (1.200mm)	PTH	Round	TOP - BOTTOM	(Mixed)	-
◇	1	70.87mil (1.800mm)	NPTH	Rectangle	TOP - BOTTOM	Rounded Rectangle	71.00mil (1.803mm)
▼	1	157.48mil (4.000mm)	PTH	Round	TOP - BOTTOM	Rounded	-
✪	5	160.00mil (4.064mm)	NPTH	Round	TOP - BOTTOM	Rounded	-
	881 Total						

Slot definitions : Routed Path Length = Calculated from tool start centre position to tool end centre position.
Hole Length = Routed Path Length + Tool Size = Slot length as defined in the PCB layout

Drill Table
FOR PTH +/- 3MIL
FOR NPTH +/- 2MIL
FOR 7.874MIL DRILL +0/-7.874MIL
FOR 8MIL DRILL +0/-8MIL
FOR 12.1MIL DRILL +0/-12.1MIL
FOR 12.2MIL DRILL +0/-12.2MIL

Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Hole Length	Routed Path Length
O	529	5.90mil (0.150mm)	PTH	Round	TOP - L2	-	-
	529 Total						

FOR 5.9MIL DRILL +0/-5.9MIL

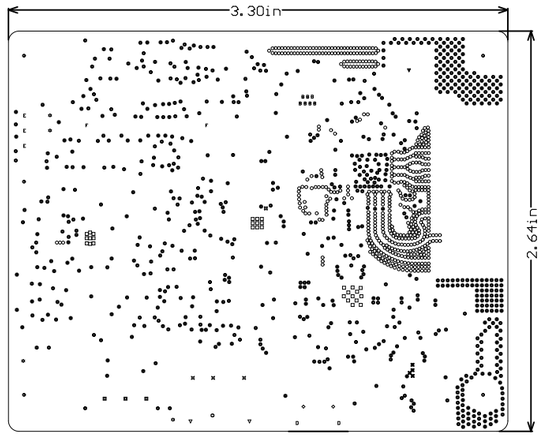
L1-L2
L1-L6

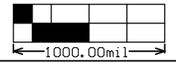
LAYER STACKUP :

Layer	Name	Material	Thickness	Constant	Board Layer Stack
1	Top Overlay				
2	Top Solder	Solder Resist	0.80mil	3.5	
3	TOP	Copper	1.60mil		
4	Dielectric1	Rogers 4835	4.00mil	3.66	
5	L2	Copper	1.20mil		
6	Dielectric2	370HR Prepreg 1080	5.70mil	3.9	
7	L3	Copper	1.20mil		
8	Dielectric3	370HR Prepreg 1080	28.00mil	4.36	
9	L4	Copper	1.20mil		
10	Dielectric4	370HR Prepreg 1080	5.84mil	3.9	
11	L5	Copper	1.20mil		
12	Dielectric5	370HR Prepreg 1080	4.00mil	4.26	
13	BOTTOM	Copper	1.60mil		
14	Bottom Solder	Solder Resist	0.80mil	3.5	
15	Bottom Overlay				

IMPEDANCE TABLE :

LAYER NO	CONDUCTOR WIDTH	DIFF. PAIR SPACING	CPWG SPACING	IMPEDANCE +/-10%	REFERENCE GND
1 (TOP)	5.2mil	5mil	N/A	100ohm	L2-GND





NOTES: UNLESS OTHERWISE SPECIFIED.

1. FOR BOARD THICKNESS AND IMPEDANCE DETAILS REFER STACKUP DOCUMENT.
2. ALL VIAS ON PAD INCLUDING BGA AREA SHOULD BE FILLED WITH NON CONDUCTIVE EPOXY AND SURFACE SHOULD BE FLAT
   THERE ARE MANY VIAS ON THE PCB WHICH ARE EITHER ON THE PADS OR CLOSE TO PADS
   THESE NEED TO BE FILLED WITH A NON-CONDUCTIVE EPOXY TO ENSURE FLAT SURFACE.
   BGA AREA VIAS SHOULD BE CAPPED WITH COPPER PLATING TO ENSURE FLAT SURFACE
   FLATNESS TOLERANCE FOR VIA ON PADS: +0.000 /- 0.001 INCHES ON BOTH SIDES.
3. SOLDER MASK OPENING IS KEPT SAME SIZE AS PAD (1:1)
   THE MANUFACTURER IS REQUESTED TO RESIZE IT AS PER THEIR SOLDERMASK TOLERANCE EXCEPT NPTH DRILL AND FIDUCIALS.
4. VENDOR SHOULD FOLLOW ROHS COMPLIANT PROCESS AND Pb FREE FOR MANUFACTURING
5. MANUFACTURER'S IDENTIFICATION,DATECODE LETTER SHALL BE SILKSCREENED ON SOLDER SIDE
   OF THE BOARD.
6. TRACE WIDTH SHOULD BE ACCURATELY ETCHED. MAX TOLERANCE +/- 1 MIL
7. BOARD DIMENSIONS ARE IN (INCHES).
8. BOW AND TWIST SHALL NOT EXCEED 0.7% OF LONGEST SIDE
9. LAYER TO LAYER REGISTRATION SHALL BE WITHIN +/-2 MIL
10. FOR ANTENNA ETCHING TOLERANCE REFER "ANTENNA ETCHING REQUIREMENTS" DOCUMENT.

DESIGN INFORMATION

MIN. TRACK WIDTH: 4 MIL
MIN. CLEARANCE: 3 MIL
MIN. VIA PAD SIZE: 12.6 MIL
MINIMUM ANNULAR RING 0.05mm (2MIL) EXTERNAL
PER IPC-D-275 CLASS 2 LEVEL C
REGISTRATION TOLERANCES: METAL +/- 2 MIL, HOLES +/- 3 MIL
HOLE SIZE TOLERANCE (UNLESS OTHERWISE SPECIFIED): +/- 3 MIL

MATERIAL:
☐ FR-408 ☐ FR-4 High Tg ☒ OTHER REFER STACKUP
THICKNESS: ☐ 56.214 MIL(1.42mm) +/-10% ☒ OTHER 57.14 MIL +/-10%
TOLERANCE: ☒ ANSI IPC-6012 TYPE 3 CLASS 2
☐ OTHER +/- _____
BOW & TWIST: ☒ ANSI IPC-6012 TYPE 3 CLASS 2
☐ OTHER +/- _____

DRILLING:
REFERENCE: ☒ AS SHOWN ☒ NC_DRILL FILES
PTH COPPER THICKNESS: ☒ 20-30 um ☐ OTHER _____

BOARD FINISH:
SILKSCREEN: ☒ TOP ☒ BOTTOM
SILKSCREEN COLOR: ☒ WHITE ☐ OTHER _____
SOLDER RESIST COLOR: ☐ GREEN ☒ OTHER RED
☒ MATTE ☐ SEMI-GLOSS

SURFACE FINISH: ☐ IMMERSION GOLD (ENIG) ☐ ENEPIG
☐ IMM. TIN/SILVER OR EQUIV ☒ OTHER IMMERSION SILVER

ARRAY/PANEL: ☒ CUT AND TRIM PER M1 BOARD OUTLINE
☐ N.C. ROUTE ☐ V. SCORE

CERTIFICATION: MATERIALS AND WORKMANSHIP FOR ALL PCBS TO MEET OR EXCEED THE REQUIREMENTS OF:
☒ ANSI IPC-A-600F CLASS -> ☐ 1 ☒ 2 ☐ 3
☐ RoHS ☐ OTHER PER ORDER

ALL BOARDS MUST MEET OR EXCEED UL94-V0 REQUIREMENTS.
PCB MUST BEAR THE UL94V-0 UL REGISTERED MATERIAL ID NUMBER

ADDITIONAL REQUIREMENTS:
MICROSECTION: ☐ YES
BARE BOARD ELEC. TEST: ☐ NONE ☒ REQUIRED ☐ PER ORDER
☐ XX MIL VIAS REQUIRE NON-CONDUCTIVE FILL AND PLANARIZE
☐ XX MIL VIAS REQUIRE CONDUCTIVE FILL AND PLANARIZE
☐ OUTER XX MIL TRACES REQUIRE 50 OHM SINGLE-ENDED IMPEDANCE
☐ LAYER 2 & 3 (INNER LAYERS) XX MIL WIDE, XX MIL SPACE TRACES REQUIRE 100 OHM DIFFERENTIAL IMPEDANCE

TEXAS INSTRUMENTS

PROJECT TITLE: xWR1843EVM
DESIGNED FOR: Public Release
FILE NAME: PROC051C_PCB.PcbDoc

ALL ARTWORK VIEWED FROM TOP SIDE	BOARD #: PROC051	REV: C	SVN REV: Not In VersionControl
LAYER NAME = [illegible]	TID #: N/A		
PLOT NAME = Fabrication Drawing	GENERATED : 3/12/2019 10:55:45 AM		TEXAS INSTRUMENTS

Texas Instruments (TI) and/or its licensors do not warrant the accuracy or completeness of this specification or any information contained therein. TI and/or its licensors do not warrant that this design will meet the specifications, will be suitable for your application or fit for any particular purpose, or will operate in an implementation. TI and/or its licensors do not warrant that the design is production worthy. You should completely validate and test your design implementation to confirm the system functionality for your application.

ENGINEER:	LAYOUT BY:
Adrian Ozer	Adrian Ozer
SCALE: 1.00	ALTIUM DESIGNER VERSION: 17.1.5.472